Defense Technical Information Center
Compilation Part Notice

ADP014178

TITLE: Experimental Aspects of Code Verification/Validation Application to Internal and Afterbody Aerodynamics

DISTRIBUTION: Approved for public release, distribution unlimited
Availability: Hard copy only.

This paper is part of the following report:

TITLE: Reduction of Military Vehicle Acquisition Time and Cost through Advanced Modelling and Virtual Simulation [La reduction des couts et des delais d'acquisition des vehicules militaires par la modelisation avancee et la simulation de produit virtuel]

To order the complete compilation report, use: ADA415759

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP014142 thru ADP014198

Experimental Aspects of Code Verification/Validation Application to Internal and Afterbody Aerodynamics

Richard Benay, Bruno Chanetz and Jean Délery
Fundamental/Experimental Aerodynamics Department
Onera-Meudon Centre
92190 Meudon
France

Abstract

The past 40 years have known a spectacular development of CFD capabilities. It is now possible to compute complex three-dimensional unsteady flows even at the design stage by solving the Unsteady Averaged Navier-Stokes Equations (URANS approach) and progress are made every day in still more advanced approaches such as LES and DNS. However, the confidence in CFD methods is still limited because of uncertainties in the numerical accuracy of the codes and of the inadequacy of the turbulence models they use. Thus, there is still a need for well made and well documented experiments to validate the codes and to help in their improvement. Such experiments must also fulfill quality criteria to be considered as safe enough and really useful for code validations. The paper presents a discussion of the strategy to be followed to ensure the reliability and accuracy of a code by placing emphasis on the experimental aspects of code validation. The purpose is illustrated by considering recent examples of CFD validation operations based on basic - or building block - experiments. The first case considers an experiment on a purely laminar shock wave/boundary layer interaction used to assess the numerical accuracy of several codes. Other examples deal with the crucial problem of the validation of turbulence models in strongly interacting flows. The conclusion stresses the importance to constitute high quality data banks on typical flows still difficult to predict. The problem of data dissemination is also briefly addressed.

1.   INTRODUCTION

Formerly, validation of predictive methods was made by comparison of the computed results with some measured wall properties, essentially the pressure. In many situations, this kind of comparison was sufficient since "old" predictive methods, which were either fully empirical or based on a multi-component type approach, allowed only the prediction of the wall properties (pressure, skin-friction, heat-transfer) and the global performance of the vehicle. They could also give a gross idea of the flow organization by predicting, for example, the size of a separated region and the location of a separation point, but this information was more or less considered as qualitative. The flow prediction landscape has completely changed with the advent of theoretical models based on the solution of the Navier-Stokes equations. It is clear that this approach is the only suitable to compute complex flows containing shock waves, centered expansion waves, separated regions, shear layers, etc... the dissipative region being turbulent in nearly all the practical situations. Then, not only the wall properties are computed but also field quantities including the mean velocity and the turbulent quantities. However, in its present state the Navier-Stokes approach is still far from being free of critics, difficulties

persisting both on the numerical side and in the physical modeling, in particular of turbulence. There is thus a strong need to validate Navier-Stokes codes before their routine use for design purposes.

Though the prediction of wall properties remains a key target for most predictive methods, since the drag and the thermal loading are the quantities of most practical interest, it rapidly became obvious that a comparison restricted to wall properties was insufficient to properly validate the most advanced predictive methods. In general, Navier-Stokes codes give a faithful and impressive picture of the flow field structure. For example, in propulsion applications, the complex organisation of the jet with its pseudo-periodic pattern of shocks and expansion waves, the separated zone forming inside an overexpanded nozzle or on the afterbody in case of jet pluming are most often well reproduced. However, a more careful analysis of the data shows that the situation is far from being entirely satisfactory.

Firstly, it is observed that a fairly good prediction of the wall pressure can coexist with a poor quantitative prediction of the velocity field. Frequently, the extent of the separated region is underestimated, sometimes considerably. In addition, the turbulent quantities are poorly predicted, especially if the flow is separated. Such discrepancies render suspect the validity of the code since they are indicative of some basic deficiency, either in its numerical scheme or its turbulence model, or both.

Secondly, a rather fair prediction of the flow field can be accompanied by large errors in the calculation of surface properties affecting mainly transfer coefficients such as skin-friction and heat transfer.

Lastly, in certain applications, the knowledge of the outer field itself is of prime interest. This is the case for problems dealing with infrared signature where detailed description of the hot propulsive jet, with an exact localization of the Mach discs, is essential. Pollution studies necessitate a good prediction of the jet properties to allow accurate evaluation of chemical processes and species concentration. A good representation of the flow resulting from interference between the external stream and the propulsive jet(s) also requires a faithful prediction of the flow field. This is also the case of shock intersections which can be of prime importance for the performance of air-intakes at high Mach number.

The problem of code validation is still more stringent in three-dimensional applications where the Navier-Stokes approach becomes mandatory. Due to the complexity of such flows, it is clear that the consideration of the surface pressure alone is completely inadequate because this information gives a very partial view of the flow (in three-dimensional flows, it is no longer possible to infer separation from an inspection of the wall pressure distributions).

In these conditions, the validation of advanced computer codes requires well documented experiments providing detailed and reliable flow field measurements. The progress accomplished in the measurement techniques over the past 40 years, mainly with the advent of laser based optical methods, has operated a true breakthrough in our capacity to investigate complex turbulent flows, containing shock waves, strong expansions, thin shear layers and recirculating regions. We therefore possess now the necessary technical means for performing such investigations.

This paper deals with the global problem of code and models validation by comparison with experiments. After purely numerical tests, this step is essential for determining the degree of reliability of a code using a given model. Two parts can be distinguished:

➢ A first part is devoted to the strategy for code verification and validation with emphasis on the experimental facet of this action. This strategy is presented in the optics of a physical approach of the problem in which one focuses on the prediction of the flow fundamental properties. The performance aspect is not considered, although it is the end product of the validation chain; this ultimate step is more in the hands of engineers than scientists.
➢ In a second part, the verification/validation strategy is illustrated by examining problems met in propulsion aerodynamics. They concern phenomena of internal aerodynamics, affecting supersonic air intakes, base flows and propulsive after-body recently studied at Onera.

2. THE CODE VERIFICATION/VALIDATION PROCESS

2.1 What is required from the code

Before considering a verification/validation action, first of all, the aim of the calculation must be clearly stressed and identified.

➢ If calculation is used to predict the performance of a system or a sub-system, accuracy is mandatory since then the engineer has to rely on the calculation to define the object which must satisfy prescribed specifications or to evaluate the aerodynamic performance achieved by a given object, in terms of drag, thrust, maximum range, stability, fuel consumption, manufacturing and maintenance costs, etc . This consideration applies to an airfoil, a wing, an air-intake, a nozzle, a propelled afterbody, etc. Such aerodynamic calculations are frequently coupled with structural analysis, aeroelasticity, thermal calculations, flight mechanics.
➢ In the design of machines involving complex flows whose experimental simulation is difficult, if not impossible, a calculation showing the flow field organization is of great help for the designer. In this case, accuracy is not essential, but reliability is crucial since one must be confident on the physical features of the computed field. This applies to flow in turbomachines where a detailed experimental description of the flow is still largely out of reach. This is also true for the flow past multi-body space launchers where information on the main flow features can be useful to detect possible hazards and to take precautions to avoid them.
➢ From a fundamental point of view, the physical understanding of complex flows has to be based on a parallel theoretical analysis whose aim is to help in the interpretation of the phenomena and to establish the descriptive approach on firm rational arguments. In this case, accuracy is not needed since, for example, theoretical analyses are nearly always derived from more or less strong simplifying assumptions rendering quantitative results questionable. This is for example the case of stability analyses, perturbation methods or asymptotic expansion techniques. These theories have greatly helped in the understanding of critical phenomena, in spite of a relatively poor quantitative agreement. Numerical calculations are also a precious tool to construct the structure of complex flows by providing information on "hidden" faces of the phenomena which cannot be reached by direct observation. This point concerns for example the topological construction of three-dimensional separated flows. Here accuracy is not essential; even reliability can be limited

since in the process there is a continuous exchange between experiment and calculation - or theory - which allows a cross fertilization of the two approaches.

➢ In the last issue, a code is used as a tool to test a new physical model (or to introduce some improvement in an existing one). Then, numerical accuracy is mandatory since it would be vain to implement a good turbulence model (if there is one) in an inaccurate code in which high gradient regions where turbulent is at work are not correctly captured.

2.2 The different steps of the validation

Considering the above points, it appears that the verification/validation procedure has to be submitted to a four step strategy or tactical actions.

First step : assessment of the code numerical accuracy : Before going into a more involved operation of code validation, encompassing all the aspects of the computing action, it seems obvious to first establish the accuracy and reliability of the code by focusing on its numerical aspects. A clear assessment of this point is not a straightforward issue in the sense that the numerics involves several closely linked aspects. Assessment of the code numerics can be done first from comparison with exact analytical solutions or with well established empirical results. Most often these exact solutions are only available for laminar flows.

In a second phase, verification can be made by confrontation with other codes developed by independent teams. This action implies a close co-operation between the persons involved in the procedure with a complete exchange of information on the calculation methods and a common evaluation of the results. A precaution for this action to be conclusive should be to agree on exactly the same configuration and to insure spatial convergence in all the calculations. If the comparison between the codes exhibits differences which cannot be resolved after a thorough analysis of the results, then one has to refer to the verdict of an independent authority which can be experiment.

A further - and hopefully conclusive - step will be to run the competing codes on a configuration for which good and reliable experimental results exist. This point is far less obvious that it would appear at first sight, since the experimental data should allow to draw clear conclusions. Thus, one should avoid to mix modeling and numerical problems by considering a simple laminar case, in a calorically perfect gas in order that the basic flow physical properties (thermodynamics constants, molecular viscosity) are well known. This point is far from being easy to achieve since it is most often very difficult to maintain an entirely laminar regime in classical aerodynamics.

The geometry of the body should be simple and completely defined to avoid complex meshing problems. From this point of view, a two-dimensional case is preferable but, as it will be seen below, an axisymmetric configuration is preferable. Experimental requirements will be exposed more completely below.

Second step: Validation of the physics implemented in the code on elementary configurations. This is the most important point for the specialist in flow physics, the first step being only a preliminary step simply aiming at verifying the tool. In the second step, the code is used to compute what can be considered as the elementary components of an aerodynamic flow: attached boundary layer, laminar-turbulent transition on a flat plate, separation induced by an obstacle, flow past a base, shock wave/boundary layer interaction, start and development of a vortex structure, vortex breakdown, shock/shock interference or shock crossing, etc. Two-dimensional - preferably axisymmetric - as well as three-dimensional basic situations have to

be considered. For this first validation step, the numerical results are compared with *building block* experiments focusing on a specific elementary phenomenon.

Third step: validation on more complex sub-systems. Once the code and its physical model(s) have been validated on basic cases, a more complete configuration must be considered consisting in a sub-system of a complete vehicle, where several elementary phenomena are combined. This is the case of a profile on which one encounters laminar-turbulent transition, attached boundary layers, transonic shock wave/boundary layer interaction, separation, wake development, etc. The wing constitutes a three-dimensional extension with the additional problems of the vortices emanating from the wing and control surfaces extremities. The supersonic air-intake involves shock/shock interference, shock wave/boundary layer interactions, corner flows with vortex development. After-bodies combine supersonic jets with complex shock patterns (Mach disc formation), shock induced separation, either inside the nozzle (overexpanded nozzle) or on the fuselage (jet pluming at the exit of an underexpanded nozzle). Many other examples can be cited: propulsive nacelle, compressor/turbine cascade, helicopter rotor, etc.

Fourth step: validation on the complete vehicle or object. This is the ultimate target in which the code is applied to a complete airplane, an automobile, a space launcher, a helicopter, etc. This part of the exercise is the domain of engineers and will not be developed here.

2.3 Requirements for good test cases constitution

We are thus naturally led to define what are the requirements for really useful experimental results and ask the question: is the data bank safe? This is a vital issue on which we will concentrate since the points which follow are the every day concern of experimentalists.

Definition of the geometry. A first condition for any experiment aiming at the verification of the numerical accuracy of a code and the validation of its physical models is to focus on a configuration whose geometry must be both representative of a typical situation, precisely defined and as simple as possible. A combination of plane surface (like a two-dimensional ramp) is a good choice but one should carefully avoid special situations, like a sharp leading edge, leading to singularities and to meshing difficulties. When possible, an analytical definition of the contour should be provided. It is preferable to give the dimensions in metric units to avoid risk of confusion in the reference length used to compute a Reynolds number. When possible, a two-dimensional geometry should be adopted - even for three-dimensional problems - since it offers many advantages to visualize the phenomena and to execute measurements, in addition of the lower cost of the test-set up fabrication. Furthermore, the original set-up must frequently be modified before arriving at a fully satisfactory flow; such modifications are far easier on a two-dimensional arrangement.

Boundary conditions. The boundary conditions must be well identified and accurately known. This concerns the upstream flow conditions (Mach number, velocity, pressure, density) when a uniform incoming flow exists. In transonic experiments executed in a channel type arrangement, one often considers phenomena taking place on the channel walls, the test section itself being the model. In this case, a well defined origin with a uniform state at upstream infinity does not exist. Then, the data should provide all the flow conditions in a section located sufficiently far upstream of the region of interest, including the boundary layer properties (mean velocity profile, turbulent quantities).

In all cases the stagnation conditions (pressure, temperature) and the incoming stream thermodynamic properties must be given. Downstream boundary conditions leading to a well

posed problem must be provided. If the flow leaving the zone of interest is supersonic, then no-conditions have to be imposed to perform the calculation. The question of the downstream conditions is more delicate if the configuration is such that the flow leaving the test region is subsonic. When the downstream flow is again uniform, most often a downstream pressure is given, since this quantity is easily obtained. It is far more difficult to provide the pressure field in a complete plane, as some theoreticians sometimes ask for. In transonic channel experiments where a shock is produced by the choking effect of a second throat, the best way is to provide the geometry of the second throat and, in the calculation, to impose downstream conditions insuring the choking of this throat.

Perturbing effects. Side effects or uncontrolled perturbations must be avoided, except if they can be taken into account by the calculation. The side effects due to the finite span of any experimental arrangement strongly affect the flow when separation occurs. Then, the experimented flow can be very different from the assumed ideal two-dimensional flow which would correspond to the infinite span condition. Confrontation of such an experiment with a planar two-dimensional calculation can be deprived of any signification and lead to entirely erroneous conclusions. Much has been said and written about the side effects to arrive at the conclusion that comparison of a two-dimensional calculation with a "pseudo" two-dimensional experiment *should be forbidden*! Even if many experiments show a limited zone in the vicinity of the wind tunnel mid longitudinal plane that can be considered as reasonably two dimensional, this is never a warranty of two dimensionality, at least locally, since the entire phenomenon can be displaced by side effects. This is the case of the separation line on profiles. If one desires to keep the mathematical simplicity of two space dimensions, the best is to compute an axisymmetric flow.

Experimental needs. The description of the flow must be as complete as possible in order to permit a thorough validation of the code and to provide all the information useful to understand the physics of the flow and to help in the elucidation of causes of disagreement. This concerns wall quantities, like pressure, heat transfer, skin-friction, field quantities such as stagnation pressure and temperature, mean velocity, Reynolds tensor components, density, species concentration, etc. Flow visualizations are highly desirable to give a precise idea of the flow field structure: shock waves, location of shear layers, separated region, vortices. A really complete description of the flow is never possible because the experiment cost would become incompatible with the budgets currently allocated to fundamental aerodynamics.

Measurements reliability and accuracy. The experimental data must be reliable. This means that the experiment is not "polluted" by an extraneous phenomenon due to a bad definition of the test arrangement or to a bad functioning of the facility. The general regulation tends now to impose to experimentalists precise information on the uncertainty margins of their measurements. This information, although important for an in depth validation of codes, is not always as essential as claimed if the objective is to test the physical validity of a complex model. The problem is different in the case of performance determination, where quantities like lift, drag, efficiency must be known with high accuracy.

The physical interpretation. Constitution of safe data banks is not restricted to the execution of hopefully good experiments in relation with code development. The experimentalist must also be a physicist able to interpret its findings and to understand the physics of the investigated flow field. This interpretation, which must be based on theoretical arguments, is essential to insure the safety of the results. It must precede any numerical exploitation.

3. APPLICATION OF THE STRATEGY TO INTERNAL AERODYNAMICS AND AFTERBODY FLOWS

3.1 Scope of the action

The propulsion of aerial vehicles, such as military or civil aircraft, missiles, space-launchers, involves parts where aerodynamics plays a major role, both as components participating to the engine functioning and as elements contributing to the thrust of the propulsive unit. These components, or sub-systems, are basically the air-intake, the nozzle and the afterbody. The role of the air-intake is to supply the engine with low Mach number and "good" quality air (i.e., with high stagnation pressure and low distortion). The air-intake contributes also to the thrust and is an important cause of drag (captation and cowl drag). The main roles of the nozzle is to regulate the mass flow traversing the propulsion unit and to insure an optimum expansion of the combustion gas to produce maximum thrust. The afterbody part is most often characterized by recirculating zones situated close to the walls, the pressure level in these zones determining the drag level (base drag). The aerodynamics of propulsion involves interactions between the internal flow and the outer stream which may strongly affect the propulsion efficiency. These phenomena are specially important at the nozzle exit and around the afterbody since they can lead to separation, either inside the nozzle or on the fuselage, according to the nozzle expansion ratio. Internal aerodynamics is probably a domain where major progress can be made in the coming year since it concerns flows involving strong shock waves and stream confluence's which are at the origin of shock/shock interactions, extended separations, shear layer developments, large scale fluctuations, these phenomena taking place in turbulent compressible flows (except in very high altitude flight). In addition, in most applications, the configuration is highly three-dimensional (twin-nozzle of combat aircraft, multi-body space launchers, etc.). All these points make the prediction of such flows difficult since they involve the hardest points met in applied fluid mechanics. The present predictive capacity being still limited there is a strong need to maintain a sustained research effort both on the experimental and theoretical sides for propulsion applications.

These problems are more critical for a hypersonic air-breathing spacecraft which flies in conditions where shock phenomena are much amplified leading to still more severe interactions between the internal and outer flows. Then, integration of the propulsion unit in the vehicle architecture is a vital issue.

The conception of the air-intake must take into account two major aerodynamic interactions. Firstly, compression of the capted high Mach number flow is achieved partly by taking opportunity of the shape of the vehicle front part which allows to realize a nearly isentropic compression, partly by a succession of ramps and/or shock reflections constituting the air-intake itself. The design of the compression ramp system is a complex matter combining the search for maximum efficiency, minimum cowl drag, minimum length or weight of the system, possibility of adaptation, etc. In particular, one is confronted with the risk of separation at a ramp or a shock reflection, with the subsequent loss of efficiency, possible occurrence of instabilities and air-intake unstart. This problem is not restricted to air-intake since separation is likely to occur in several other parts of the vehicle. As it is well known, separation is most often avoided by designers since it leads to a degradation of performance and a rise of the nuisance produced by the vehicle. In addition, at hypersonic velocity the reattaching shear layer of a separation bubble leads to extremely severe high level of heat fluxes. All these reasons make essential an accurate prediction of separated flows.

At the other extremity of the propulsion nacelle, one is faced with another class of phenomena resulting from the confluence between the propulsive jet(s) and the outer flow. In many circumstances, the co-flowing streams are non-adapted, which, in supersonic or hypersonic regimes, generates shock waves interacting with the boundary layers. When the flow issuing from the nozzle is at a pressure well superior to the outer pressure (underexpanded case), the strength of the interaction shock can be so severe as to force the external flow to separate. On the other hand, if the nozzle is overexpanded, the rise in pressure at its exit may cause separation in the nozzle. The non-adaptation phenomena induced by flow confluence have an influence: on the afterbody drag and thrust balance, on the aerodynamic stability and maneuverability of the vehicle, when flow instabilities occur, and on the base heat-flux levels since hot gases are present inside the separated fluid regions.

The examples which follow present typical test cases, constituted at Onera for the majority of them, to help in the development of safer and more accurate codes. These experiments are all (relatively) news, which explain that they have not yet been really exploited, except the first one which is a good example of a nice verification action involving five codes. For the other cases, only one home-made Navier-Stokes code has been run with the main objective to check the completeness and safeness of the test cases.

3.2 The cylinder-flare case for hypersonic laminar separation

Aim of the operation. The aim of the present operation was to assess the ability of numerical codes using various schemes to predict separation on a 30° axisymmetric ramp flow at a Reynolds number sufficiently low to insure laminar flow over the whole interaction domain (Chanetz et al., 1998, 1999). The action consisted in extensive comparisons between measured and calculated wall and flow field properties. Since the calculated wall quantities are particularly sensitive to grid constitution, great attention was paid to the quality of the grids. The results of five codes have been compared in this action:

➢ Two Navier-Stokes codes from Onera using a finite-volume approach. These are the FLU3M code (Borrel et al., 1988) and the Nasca code (Benay and Servel, 1995).
➢ The Navier-Stokes code HIG-2XP from the University of Rome 'La Sapienza' using a finite-volume approach (Grasso and Marini, 1996).
➢ The Navier-Stokes code FLOW from DLR using a finite-element approach (Schulte et al., 1998).
➢ A Direct Simulation Monte-Carlo code (DSMC) from NASA-Langley Research Center (Moss and Olejniczak, 1998).

This operation corresponds to step 1 of the strategy defined in Section 2.3: assessment of the code numerical accuracy.

The experimental part. The experiments have been executed in the low Reynolds number Onera R5Ch wind tunnel. For the nominal stagnation conditions (pressure $p_{st} = 2.5 \times 10^5$Pa, temperature $T_{st} = 1050$K), the upstream Mach number was equal to $M_0 = 9.91$.

The model (see Fig. 1) is constituted by a hollow cylinder, with a sharp leading edge, followed by a flare terminated by a cylindrical part. The flare angle, $\beta = 30°$, induces a large separated zone. The flare is followed by a cylindrical part in order to facilitate the computations by displacing the base flow sufficiently far downstream of the interesting area, in such a way that the complex phenomena occurring in the base region have no effect on the interaction region. The model has a total length of 170mm, the reference length, based on the distance between the sharp leading edge and the beginning of the flare being equal to $L = 101.7$mm. The Reynolds

number calculated with L is equal to Re_L = 18,375. The outer diameter of the cylinder is equal to 65m and its inner diameter to 45m.
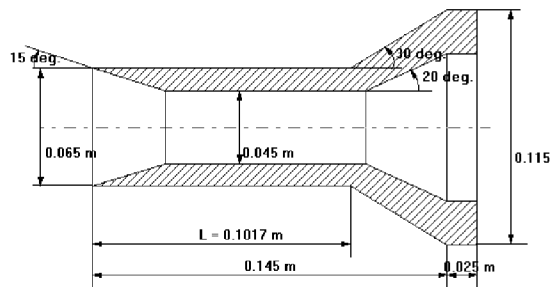

Figure 1: Geometrical definition of the cylinder-flare axisymmetric model

The flow has been qualified by surface flow visualization, wall pressure and heat transfer measurements and probing of the outer field by use of the X-ray electron beam fluorescence technique

CFD codes confrontation. As far as the three finite-volume Navier-Stokes calculations are concerned (FLU3M, Nasca and HIG-2XP), a preliminary grid dependency study by successive dichotomic mesh refinements in both directions was performed with the FLU3M code. As a result, the calculations were performed with the three codes on a (289×97) grid. For the FLOW results, the computational domain is discretized by using a hybrid grid consisting of approximately 61,500 elements (after grid adaptation). The structured subgrid situated on the isothermal model wall is composed of 40 layers, growing with geometric progression.

For the DSMC results, the calculation is made with a four-region computational domain containing 78,100 cells, where each cell is further subdivided into four (2×2 subcells). The collision partners are selected within the subcell; consequently, the flow resolution is much higher than the cell resolution (however, the microscopic properties are extracted from averages within the cell).

A comparison between the three finite-volume Navier-Stokes codes and the FLOW and DSMC codes reveals that the wall pressure distributions plotted in Fig. 2a are in good agreement with experiment in the case of the HIG-2XP and Nasca codes, as far as the prediction of separation extent is concerned. However, the pressure obtained on the flare is higher than that obtained in the experiment. The figure indicates that the HIG-2XP code is in good agreement with experimental values.

In the vicinity of reattachment, the experimental results present a non-smooth behavior. It was not possible to ascertain whether it is due to experimental uncertainties or to a real physical phenomenon (that neither of the codes capture). However, it is interesting to point out that the major discrepancies between codes in the prediction of the wall pressure on the flare start in the reattachment zone.

The examination of the skin friction coefficient distributions (see Fig. 2b) shows a good agreement between the HIG-2XP results and the experimental separation point. The locations of the separation and reattachment points are summarized in Table 1.

Abscissa(X/L)	FLU3M	Nasca	HIG-2XP	FLOW	DSMC	Experiment
Separation	0.72	0.74	0.77	0.74	0.75	0.76 ± 0.01
Reattachment	1.33	1.33	1.32	1.33	1.32	1.34 ± 0.015

Table 1 : Separation extent

Concerning the Stanton number distributions (see Fig. 2c), the Nasca and HIG-2XP codes give results on the flare showing the best agreement with experiment.
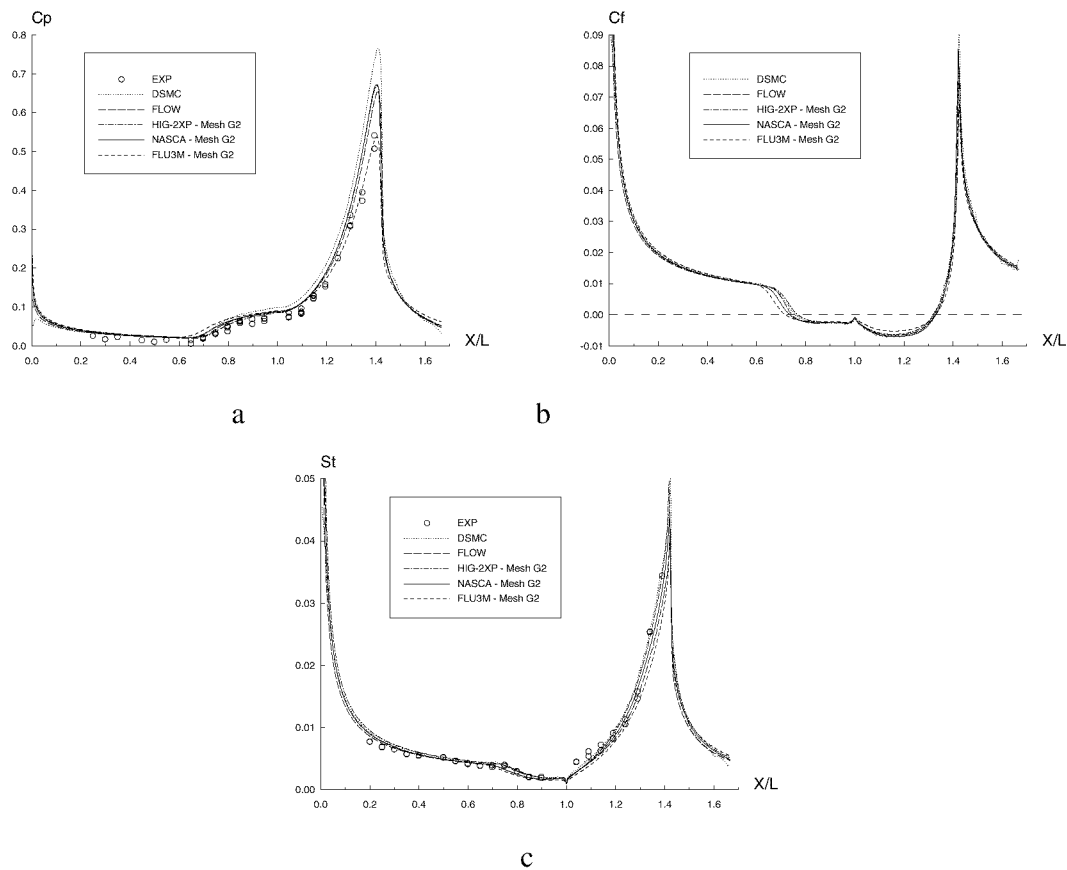

Figure 2: The cylinder flare-model at Mach 9.92 . Comparison of wall quantities
a - Pressure coefficient, b - Skin friction coefficient, c - Stanton number

Since the three finite-volume Navier-Stokes codes (FLU3M, Nasca and HIG-2XP) give nearly the same results, only three codes are considered: the finite-volume Navier-Stokes code Nasca, the finite-element Navier-Stokes code FLOW and the DSMC code.

Three density profiles have been measured by use of X-ray electron beam fluorescence (Chanetz et al.,1999). The profile at X/L = 0.3, shown in Fig. 3a, is located upstream of the separation line. At this station, the increase of density is due to the shock generated by the strong viscous interaction at the sharp leading edge. There is a good agreement between numerical and experimental results for the density peak amplitude. However, the calculated radial shock position varies with the simulation used.

At this station (X/L = 0.3), leading-edge effects probably subsist, their influence being not accurately represented by the Navier-Stokes codes. This can be due to slip effects that are not

taken into account and the difficulty to keep the correct leading edge location with the mesh in Navier-Stokes approach. The DSMC calculation is in excellent agreement with experiment. For the profiles at X/L = 0.6, shown in Fig. 3b, the inverse behavior appears on the radial shock location, the best predictions being furnished by the two Navier-Stokes codes. However, this difference on shock location needs to be confirmed before concluding on this subject. The profiles at X/L = 0.76, shown in Fig. 3c, present the same tendencies.

a   b

c
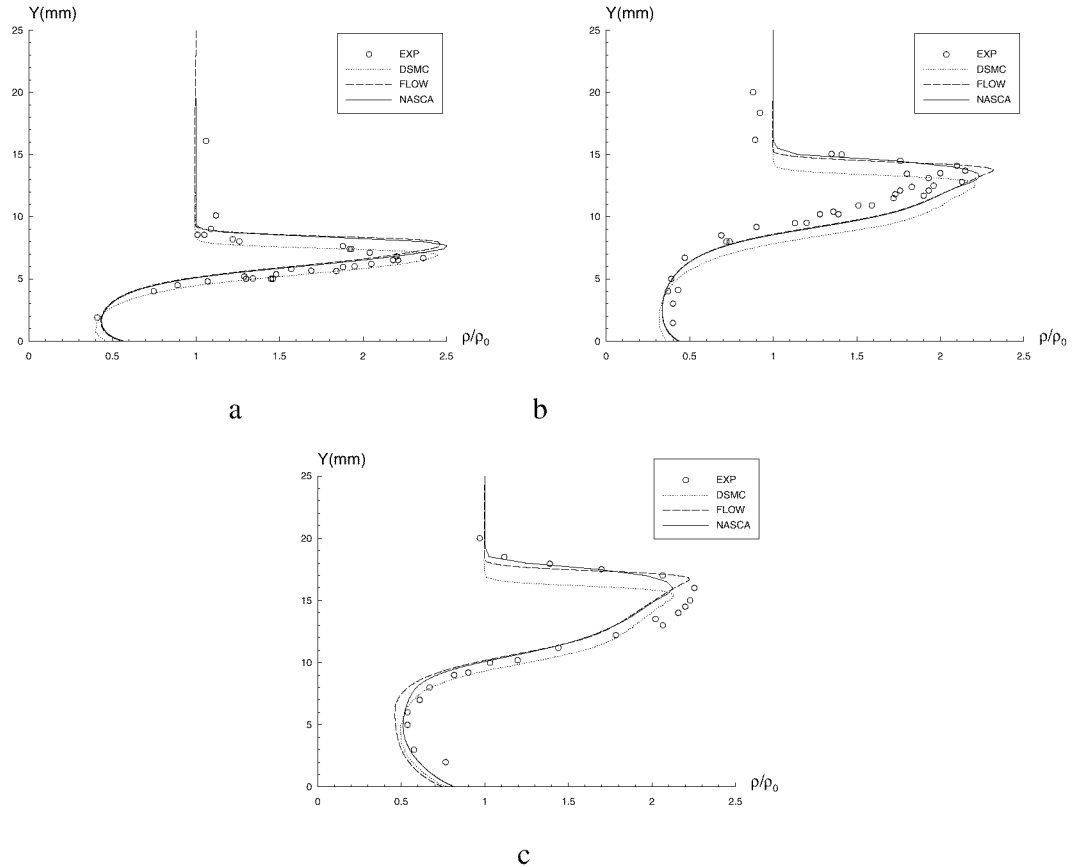
Figure 3: The cylinder flare-model at Mach 9.92 Density profiles in the interaction region. (X-Ray EBF measurements)
a - Station X/L = 0,30, b - Station X/L=0,60, c - Station X/L=076

3.3 Shock wave / boundary layer interaction control in transonic flow

Aim of the operation. Shock-waves and their interaction with the boundary-layer play a major role in determining the performance of propulsion systems such as air-intakes, diffusers, turbomachine cascade, etc. One way to reduce the harmful effects of these shocks is to perform a control action in the interaction region (Délery, 1984, Stanewsky et al., 1997). In the present study, the following techniques have been considered: 1) active control in which a part of the boundary-layer is sucked off through a slot, 2) passive control, 3) hybrid control which is a combination of a passive control cavity and a suction slot (or cavity) located downstream of it. The aim of slot suction is to swallow part of the low energetic flow close to the surface before interaction of the boundary layer with the shock or during the interaction process itself. The principle of passive control consists in establishing a natural circulation between the downstream high pressure face of a shock and its upstream low pressure face. This circulation is achieved through a closed cavity, placed underneath the shock foot region, the face in contact with the outer flow being made of a perforated plate. It has been shown

that, in very limited circumstances, passive control may produce a reduction of an airfoil drag, while postponing to higher incidences the limit of buffet onset. However, the gain being frequently problematic, it has been proposed to combine passive and active control to realize what is called hybrid control. Control methods are more likely to be used in air-intake applications to diminish stagnation pressure losses (improved efficiency) or stabilize a shock-wave.

The experimental part. These experiments were executed in the S8Ch transonic-supersonic basic research wind tunnel of the Onera Meudon Center. This facility is a continuous wind tunnel supplied with desiccated atmospheric air mainly dedicated to LDV measurements. The stagnation conditions were: $p_{st0} = 96,000 \pm 800$ Pa and $T_{st0} = 300 \pm 4$ K. A photograph of the test set-up is shown in Fig. 4. It is constituted by a transonic channel having a test section with a maximum height of 100mm and a span of 120mm. The lower wall is rectilinear and equipped to receive the control devices, the upper wall being a contoured profile designed to produce a uniform supersonic flow of nominal Mach number equal to 1.4. A second throat, of adjustable cross section, is placed in the test section outlet to produce by choking effect a shock-wave whose position, and hence intensity, can be adjusted in a continuous and precise manner. It also isolates the flow field from pressure perturbations emanating from downstream ducts, reducing unwanted shock oscillations. The two side walls are equipped with high quality glass windows to allow visualizations and LDV measurements.
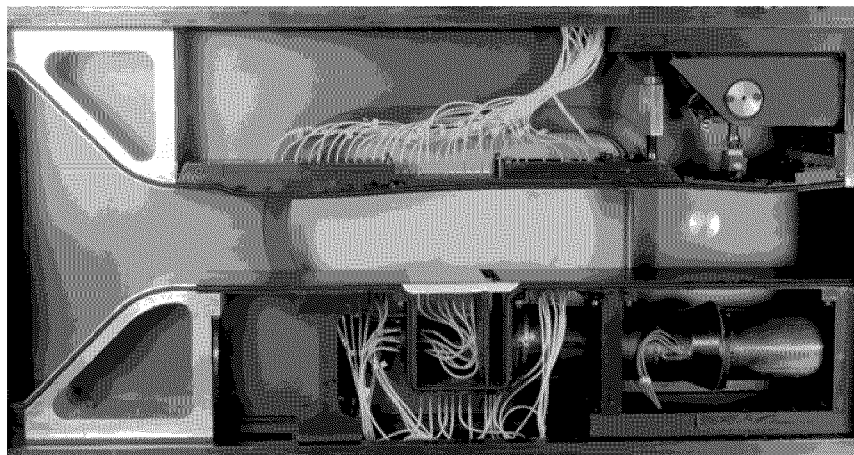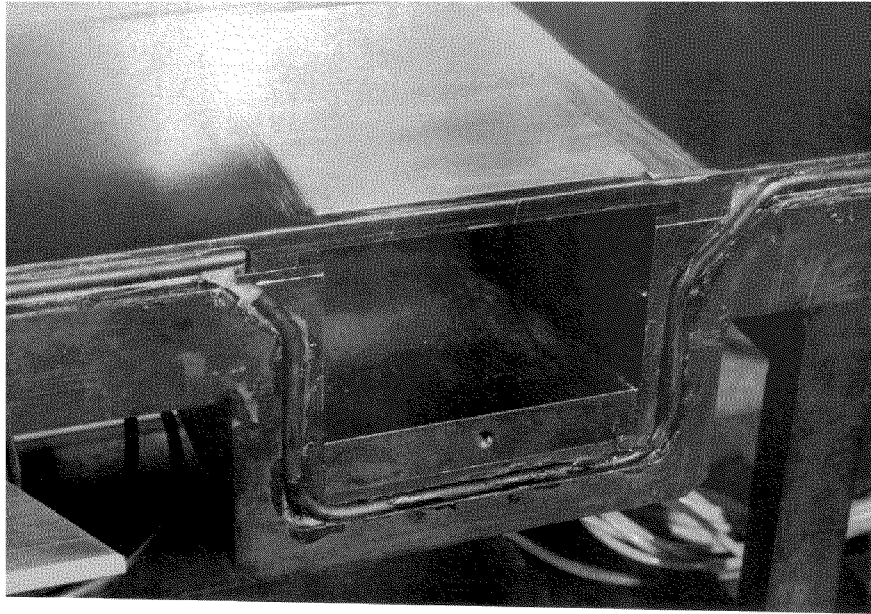

Figure 4: Shock wave/boundary layer interaction control. Test arrangement in the S8Ch wind tunnel.

The type of control taken as example here is passive control. For this device (see Fig. 5), a 70mm-long passive control cavity was used, the shock being centered on it. The cavity, which extends between X = 130mm and X = 200mm, is covered by a perforated plate, with a 5.67%-porosity and 0.3mm-diameter holes.

*Figure 5: Shock wave/boundary layer interaction control.
Arrangement for passive control*

The flows under study were qualified by schlieren visualizations and quantified by measurements of wall pressure distributions (pressure orifices being located in the vertical median plane of the test set-up) and probing of instantaneous velocity with a two-component LDV system (Bur et al., 1998).

A Navier-Stokes code confrontation. The numerical simulations were performed with the Nasca code which solves the classical Reynolds Averaged Navier-Stokes (RANS) equations. Turbulence modelling was first carried out by means of the [k-ε] transport equation models of Chien. (Ch model) (Chien, 1982) and Launder-Sharma (LS model) (Launder and Sharma, 1974). These two models were compared in the reference and passive control case with the new [k-σ] turbulence model, where σ represents a length scale (Benay et al., 2001).

The calculation domain is a part of the experimental channel extending from a well chosen section of the divergent expanding zone, where experimental velocity and turbulent shear stress profiles (obtained by LDV probing) are imposed, to the end of the channel, where the experimental pressure is imposed. Passive control is simulated by prescribing the unit mass flow ρv at the wall, the conditions on the other variables remaining unchanged. The ρv value at the wall is obtained by relations expressing a direct dependence of the wall vertical velocity to the pressure difference between the cavity and the external flow. The cavity pressure is taken as the experimental one. The relations used for the computations are the calibration law of Poll (Poll et al., 1992) and the Bohning-Doerffer law (Bohning and Doerffer, 1997).

Both transpiration/suction laws were implemented in the code and their respective results compared to experiment. The Mach number contours plotted in Fig. 6 are obtained with the [k-σ] model, the three models giving nearly identical pictures at this level. These contours show a good prediction of the large spreading of the shock system, which begins at the origin of the cavity. The difference in the location of the crossing (quasi normal) shock is weak. The thickening of the viscous zone due to the control device is well reproduced by the calculation.

Transonic interaction with boundary layer passive control
Mach number contour lines

LDV measurements

Mach: 0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1 1.1 1.2 1.3

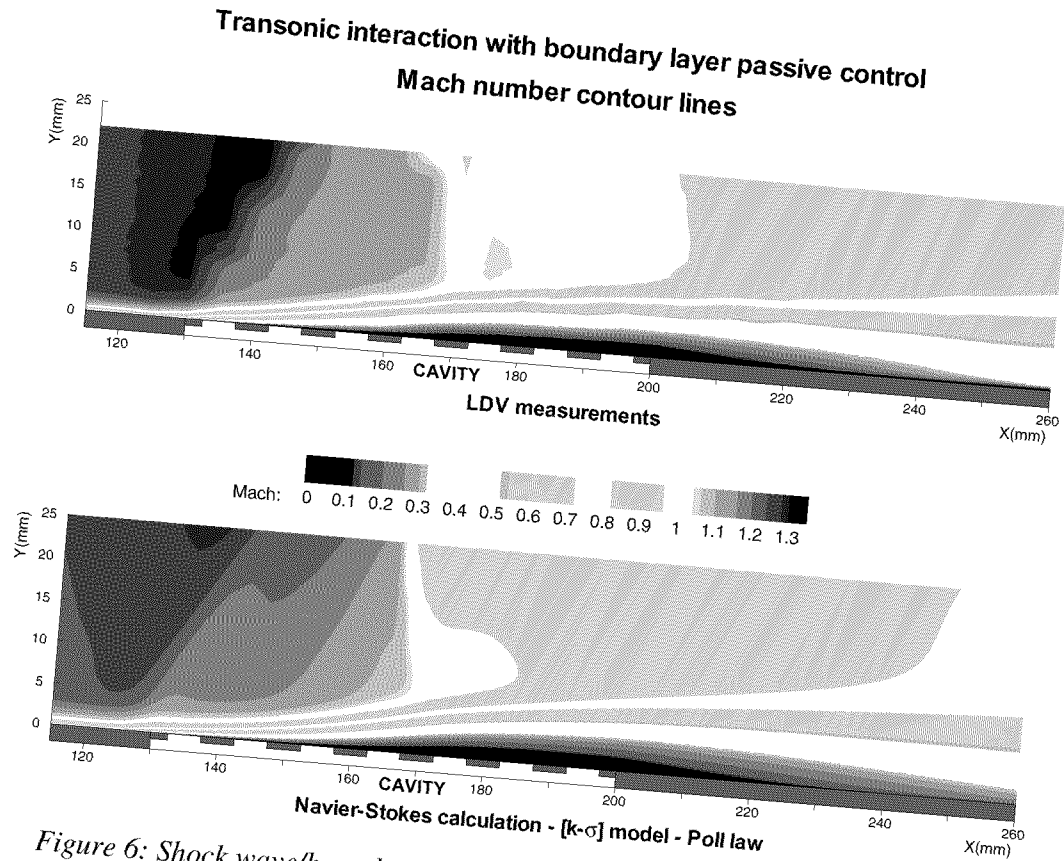Navier-Stokes calculation - [k-σ] model - Poll law

Figure 6: Shock wave/boundary layer interaction with passive control. Mach number contours: confrontation between experiment and Navier-Stokes calculation

The computed wall pressure distributions plotted in Fig. 7 show the difficulty to simulate injection through very small holes by a continuous distribution and a discrete mesh. A rigorous calculation of this problem should have been done by meshing each hole, which is unrealistic with present computing capabilities. We will see in the following results that the apparently rough approximation made to treat the porous wall condition is almost correct. At the beginning of the perforated plate (see Fig. 7), the peak on the computed wall pressure values at X = 130mm is a consequence of the sudden change of boundary condition between the two surrounding mesh points. The numerical approximation and the experimental resolution are not sufficient to give an account of the true physical process. Downstream of the mid cavity, the computed pressure recovers more satisfactory levels. A modification from the imposed experimental pressure levels on downstream boundary was necessary with the [k-ε] calculations in order to adjust the shock to its experimentally observed position. The amplitude of this shifting can be seen on Fig. 7. The agreement with experiment of computed wall pressure from the end of the cavity to the terminal downstream section is satisfactory with the [k-σ] model.
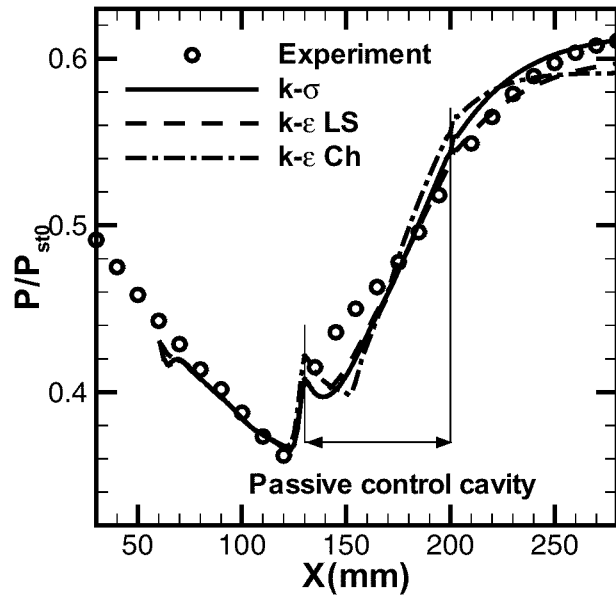
*Figure 7: Shock wave/boundary layer interaction with passive control
Computed and experimental wall pressure distribution*

As shown in Fig. 8, the streamwise velocity profile in the region of maximum wall pressure gradient, at the beginning of the cavity (X = 130mm), is submitted to the effect of the oblique thin compression fan starting from the junction between the solid and porous walls. The evolution of the boundary-layer velocity profiles at the beginning of the interaction is predicted satisfactorily by the models. An effect of passive control on the interaction is the occurrence of a separated flow of small size, above the perforated plate. As a consequence of the poor capture of the reversed flow by the models, the response of the boundary-layer flow to local compressions is too roughly simulated. This fact can be observed in Fig. 8, at X = 180mm and 220mm.
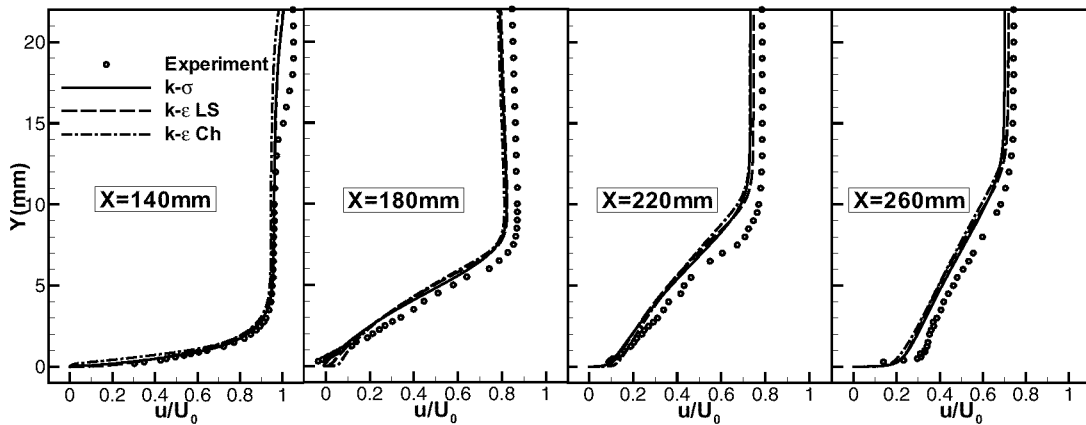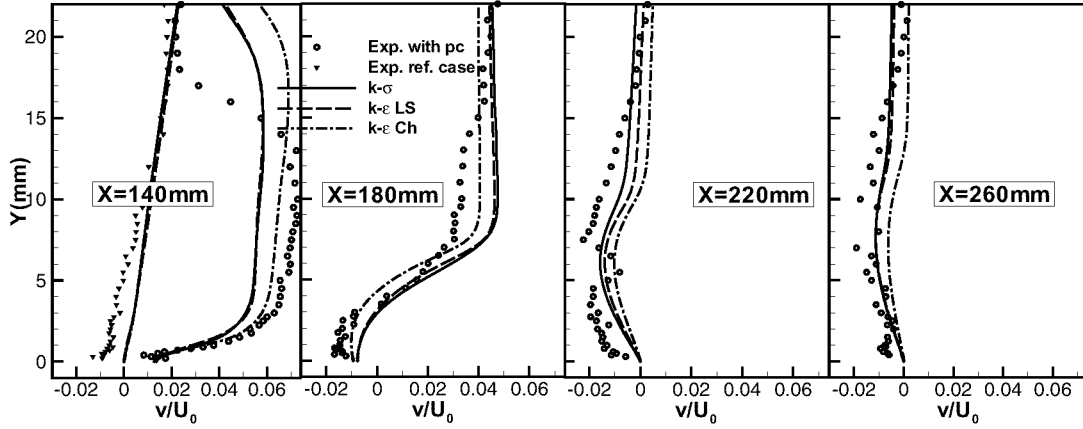

*Figure 8: Shock wave/boundary layer interaction with passive control.
Streamwise velocity profiles in the control region*

An important test for the validity of wall transpiration/suction modeling is the prediction of near-wall vertical velocity profiles. As a preliminary verification, the residual mass flow rate per unit span across the perforated plate, which should be zero, has been computed. In all the cases, the value of the ratio of this mass flow to the mass flow rate deficit $\rho_0 U_0 \delta_0^*$ in the upstream boundary-layer has been found to be lower than 10^{-5}. More detailed information is

presented in Fig. 9. Upstream of the compression system (X = 140mm), wall injection is predicted by all the models, the data at this station being compared to those obtained without control. We verify that the near wall values without injection, which should tend to zero, are affected by an error representing 1% of the upstream external flow velocity. Taking into account this experimental uncertainties, it is seen that the modeling of the passive control gives a fair prediction of the gross tendencies of the field's evolution.

Figure 9: Shock wave/boundary layer interaction with passive control.
Transverse velocity profiles in the control region

At the interaction beginning (X = 140mm), the LS and [k-σ] models anticipate the growth of the maximum $\overline{u'v'}$ level, which is not the case with the Ch model (see Fig. 10). The agreement between the LS and [k-σ] models at X = 180mm is remarkable for this controlled interaction, the Ch model strongly overpredicting the maximum level of $\overline{u'v'}$. Downstream of the interaction region, the agreement between the LS and [k-σ] models is confirmed. The too rapid near-wall variation of the turbulent shear stress level in the downstream boundary-layer characterizes the Ch model.
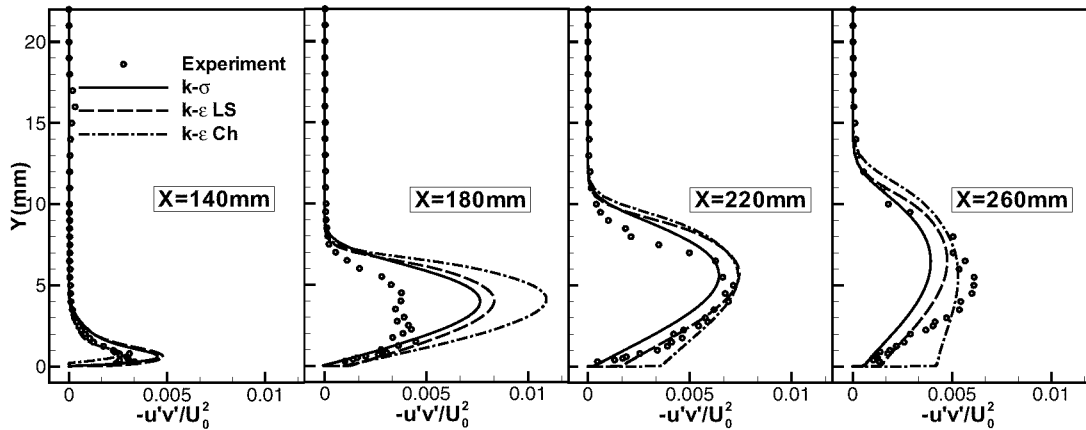

Figure 10: Shock wave/boundary layer interaction with passive control.
Cross-correlation profiles in the control region

3.4 A supersonic base flow

Aim of the operation. Validation of turbulence models requires an important number of calculations to perform the adjustment phase of the constants, which is the indispensable complement of the theoretical development of a turbulence model. Such parametrical studies are still difficult to execute with three-dimensional configurations because of the present technological limitations of the computers. Therefore, two-dimensional tests are still important and, among them, axisymmetric flows are the most appropriate configurations to obtain effective experimental two-dimensionality.

A satisfactory prediction of the turbulence evolution in largely separated flows is still not assumed by any of the presently existing turbulence models. A typical configuration where an extended zone of recirculation exists is the base flow, which constitutes an excellent test case for validating models. Base flows have been the subject of numerous experimental and theoretical studies since the fifties in order to understand the physics of such flows which are of prime importance for projectiles, missiles or space launchers. Due to the complexity of base flows, there is still a need to constitute reliable theoretical tools for predicting the drag of afterbodies and also the aerothermal loads on the base region of propelled afterbodies.

The main goal of the present study was to assess the ability of the [k-σ] two equation turbulence model to predict the mean field and some fluctuating quantities in a supersonic axisymmetric base flow. The chosen test case is an experiment executed by Herrin and Dutton (Herrin and Dutton, 1993) whose results are widely accepted for testing simulations and which has served as data base for previous theoretical studies on the subject (Sahu, 1992; Tucker and Shy, 1993; Espina and Piomelli, 1997; Fureby et al., 1999). Assessment of the model, using the Nasca code, is done by comparisons with these data and with the results given by three other well known models.

The flow field physics. The experiments of Herrin and Dutton were performed in a test facility specially designed to generate axisymmetric flows (Sauter and Dutton, 1989). In particular, profiles of mean and fluctuating velocity fields in the recirculating flow are provided by LDV measurements. The upstream Mach number has been determined from LDV (Herrin and Dutton, 1993) to be 2.46 ± 1%, while the static wall pressure measured just upstream of the base corner corresponds to a uniform flow with a Mach number of 2.44. During the present calculations we have chosen an upstream Mach number $M_0 = 2.45$.

The rapid variations of the fields and the turbulent mixing in the shear layer forming at the base shoulder are the first challenge for the modeling of the mean and fluctuating fields. Good precision in the prediction of the nearly constant pressure in the recirculating bubble, limited by the mixing layer and the rear stagnation point on the axis, is fundamental for base drag prediction. The recompression subsequent to the flow field realignment at the rear stagnation point is visible in Fig. 11, at the point where the recirculating flow progressively changes into a wake.
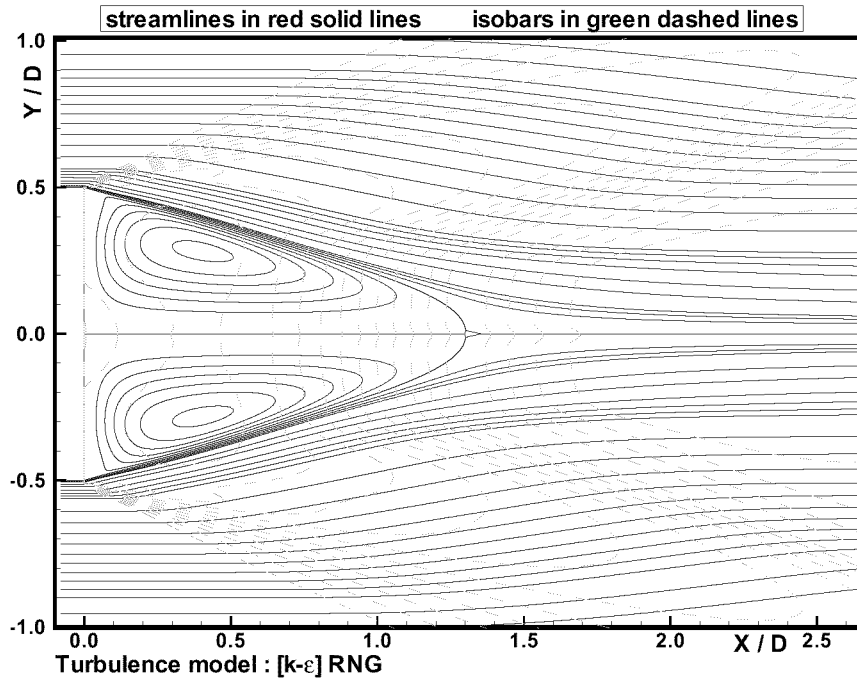
streamlines in red solid lines   isobars in green dashed lines

Turbulence model : [k-ε] RNG

Figure 11: Base flow in supersonic flow (Mach = 2.45).
Streamlines and pressure levels computed with a [k-ε] model

The experiment of Herrin and Dutton provides us with data on the evolution of the boundary layer on a restricted part of the body beginning at a distance X = -0.079D from the base (D is the cylindrical body diameter). The experimental profiles at this location must be taken as upstream boundary condition for the computations.

A Navier-Stokes code confrontation. Here, turbulence modeling was carried out by means of a [k-ε] transport equation models whose derivation is based on a renormalisation of the Navier-Stokes equations. (RNG model) (Yakhot and Orszag, 1986), the Launder-Sharma (LS model) and three versions of the [k-σ] turbulence model.

The distributions of the calculated and experimental pressure distributions on the base are shown in Fig. 12. The experimental value of the ratio p_c / p_∞ of the mean base pressure to the uniform upstream pressure is equal to 0.55. This level is 10% higher than data on base flows at the same Mach number extracted from earlier experimental compilations (Délery and Sirieix, 1979). These computed base pressure values are considerably closer to experiment than that obtained with a Baldwin-Lomax calculation (see Fig.12), in agreement with previous studies (Sahu, 1992). However, they contradict overpredictive results obtained with other two equation models (Espina and Piomelli, 1997). In a recent computation using LES (Fureby et al., 1999), a level of 0.52 for the mean base pressure was found.
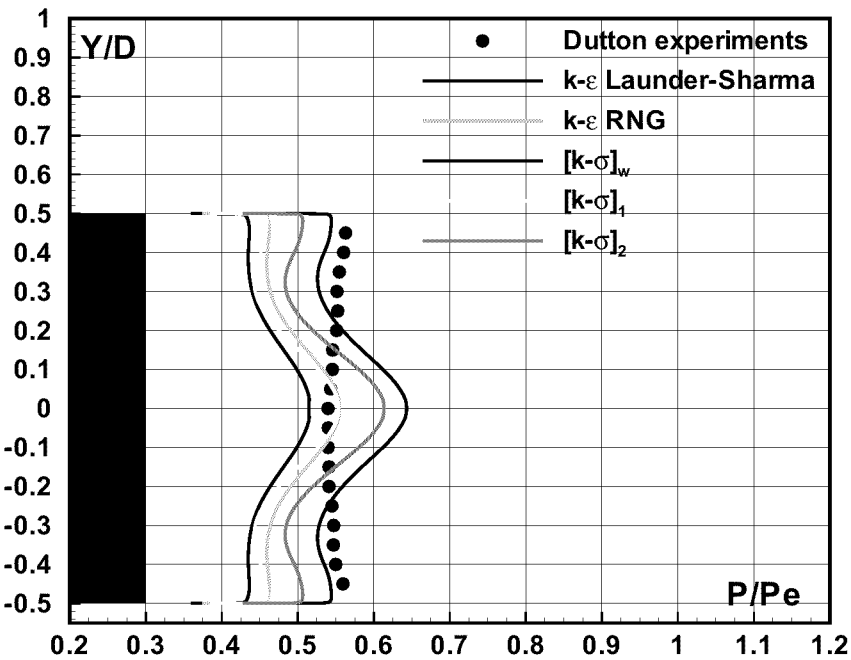
Figure 12: Base flow in supersonic flow (Mach = 2.45).
Base pressure prediction by the models

Profiles obtained in the middle of the bubble, at X = 0.63D downstream of the base, are shown in Fig. 13. A salient fact is the smoothing by the models of the shear stress evolution during the rapid transition between the mixing layer and the reversed flow. This smoothing explains the incapacity of the models to predict the nearly constant reversed flow region in the immediate vicinity of the base, the defect coming from an overprediction of the negative axial velocity on the axis. This too large radial variation of the axial mean velocity is also due to an excessive evaluation of the eddy-viscosity in the part of the bubble situated near the axis.
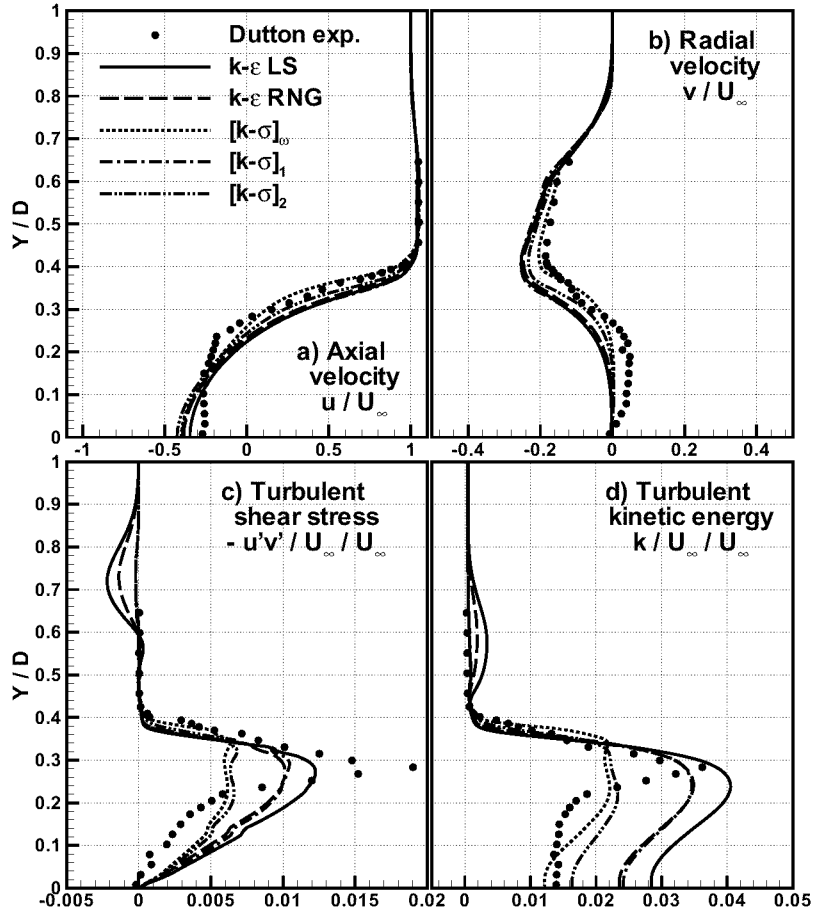
Figure 13: Base flow in supersonic flow (Mach = 2.45)
Profiles at X / D = 0.63

The results obtained by the RNG and $[k\text{-}\sigma]_1$ models are satisfactory at the rear stagnation point, located at X = 1.26D (see Fig. 14), except for the k profile. The good agreement of the axial velocity profile with experiment proves that the position of this stagnation point is well predicted by the two models.
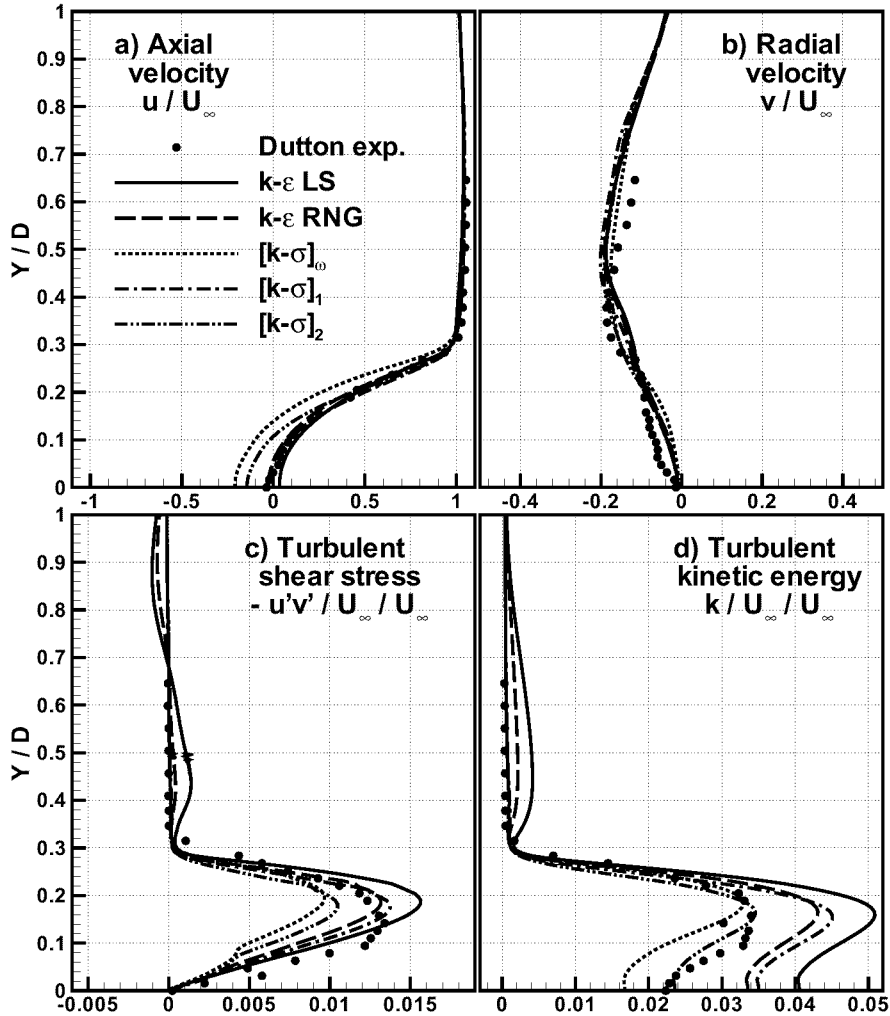
Figure 14 : Base flow in supersonic flow (Mach = 2.45)
Profiles at X / D = 1.26

3.5 Plug nozzle aerodynamics

Aim of the operation. The plug nozzle concept has been proposed in the 60s to limit thrust losses due to jet overexpansion and to confer to the nozzle some self-adaptation capabilities without having to modify its shape. In this arrangement, the supersonic expansion is realized along a center body - or plug - is place of an external contour, as in a classical nozzle. This idea has been reconsidered to equip hypersonic vehicles or space launchers having to fly in conditions strongly out of adaptation. Such nozzles can be linear or axisymmetric. In the later case, the flow coming from the engine can be ejected either through an annular throat or a series of small nozzles surrounding the plug. The present investigation has been focussed on the effects of interactions taking place between the flow produced by the nozzle and an outer supersonic stream (Reijasse and Corbel, 1997).t

The experimental part. This experiment was also executed in the S8Ch wind tunnel. A photograph of the afterbody model in the test section is shown in Fig. 15. The stagnation pressure p_{te} and the stagnation temperature T_{te} were equal to 0.99×10^5Pa and 297K, respectively. The 120×120mm^2 test section was equipped with a two dimensional nozzle

designed to give a uniform Mach number equal to 1.95. The axisymmetric afterbody model is mounted at the end of a 40mm diameter central sting fixed upstream of the nozzle throat. The diameter of the model is also equal to 40mm. The model consists of a central annular plug nozzle mounted inside of a hollow cylinder. The cylinder is terminated by a boattail and a small base. The contours of the full-length annular plug nozzle was calculated by the method of characteristics in order to provide, in the case of a perfectly adapted jet, a uniform flow with a Mach number of 3 in the section at the end of the spike. The plug nozzles was fed with desiccated high pressure air at room temperature. The jet stagnation pressure could be varied from 1 to 10×10^5 Pa.
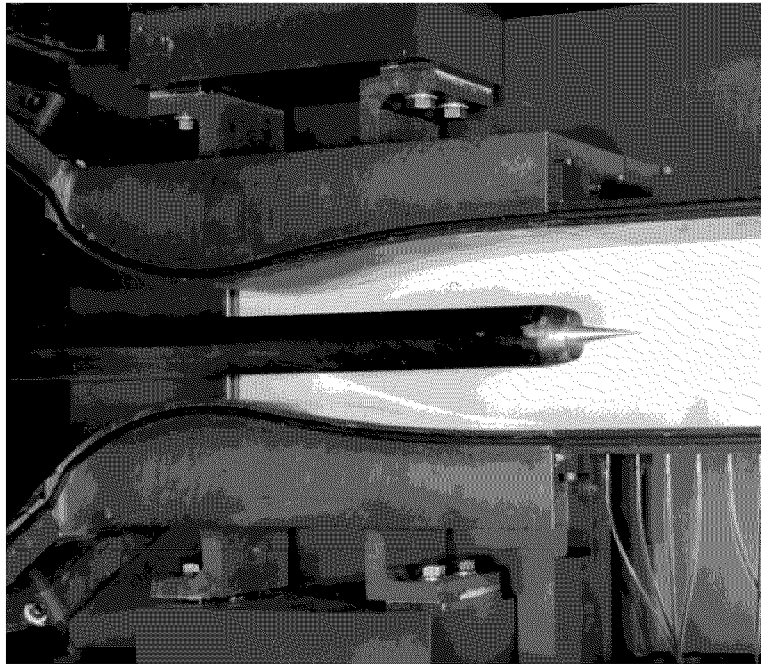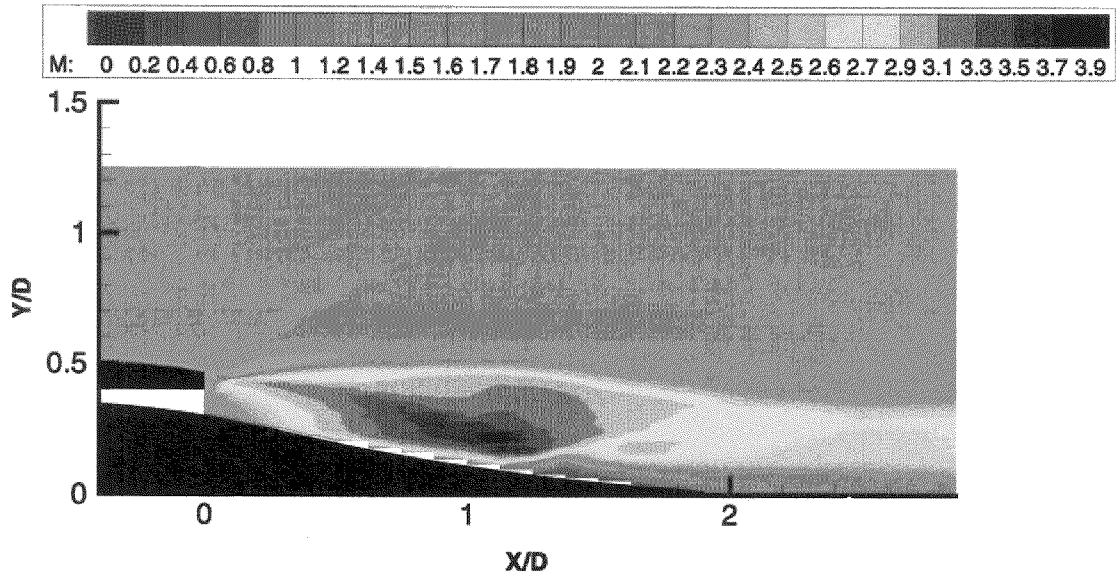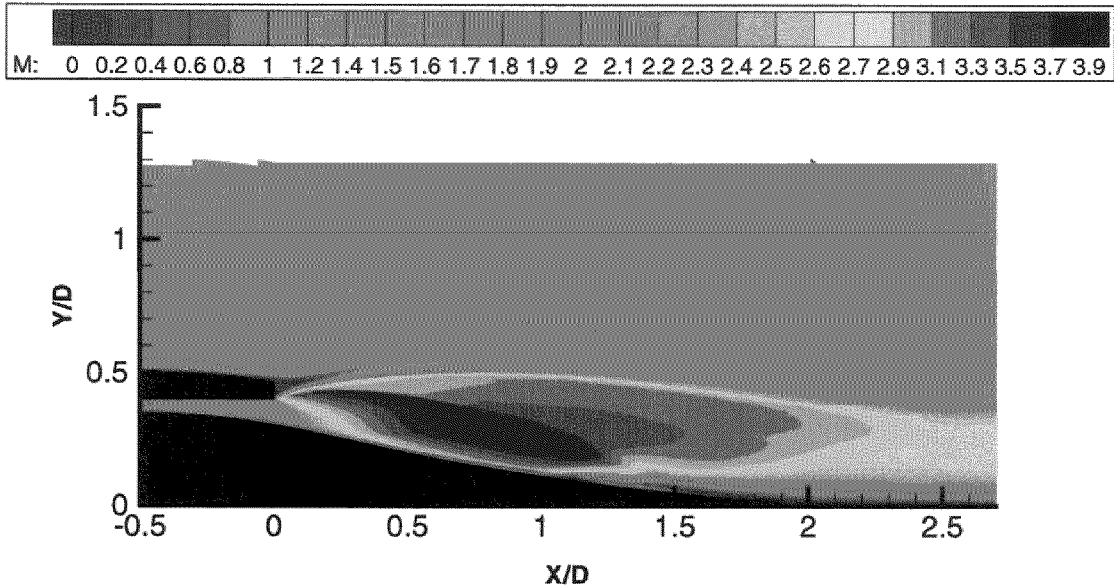

Figure. 15: Plug nozzle aerodynamics. Test set up in the S8Ch wind tunnel

The flow was qualified by means of schlieren visualisations (continuous and short exposure time), surface pressure measurements and probings in the flow vertical meridian plane with the two-component LDV system.

A Navier-Stokes code confrontation. The computed results for an expansion ratio $p_{stj}/p_{ste}=5$, have been obtained with the Nasca code by using the Launder-Sharma [k,ε] turbulence model and the [k,ε] RNG model . The comparison of the iso-Mach contours shown in Fig. 16 for the case of the Launder-Sharma model demonstrates again that a RANS calculation gives a faithful picture of the flow, although a deeper analysis of the results shows that discrepancies exist in the prediction of the strong interaction.

Figure 16: Plug nozzle aerodynamics. Mach number contours for $p_{stj}/p_{ste}=5$.
a - From LDV measurements, b - Navier-Stokes calculation

The aim of the theoretical exploitation of this experiment is to closely examine the ability of two equation turbulence models to predict mean and fluctuating fields evolutions around the spike. We take as examples, the profiles of the axial velocities, of the estimated turbulent kinetic energy and of the cross correlation u'v' on two stations of the spike situated respectively at 0.1 and 0.75 diameter downstream of the base (see on the preceding figures for these locations). The Launder-Sharma model (in red) is compared with the RNG model (in green) and with the experimental values. We see in Fig. 16, at 0.1 D, that the Launder-Sharma model predicts more satisfactorily the boundary layer developing on the strongly curved wall of the beginning spike. More downstream, at 0.75 D, we observe that the models overpredict the axial velocity (see Fig.17).
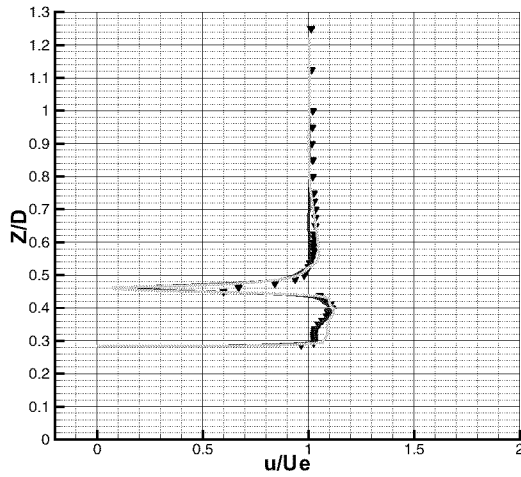
Figure 16: Plug nozzle aerodynamics
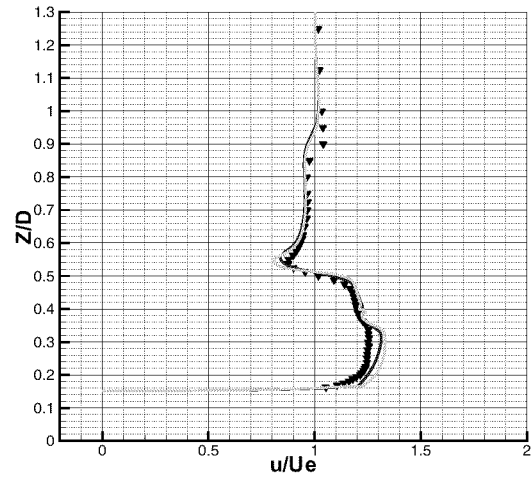Longitudinal velocity profiles at X/D=0.1

Figure 17: Plug nozzle aerodynamics
Longitudinal velocity profiles at X/D=0.75

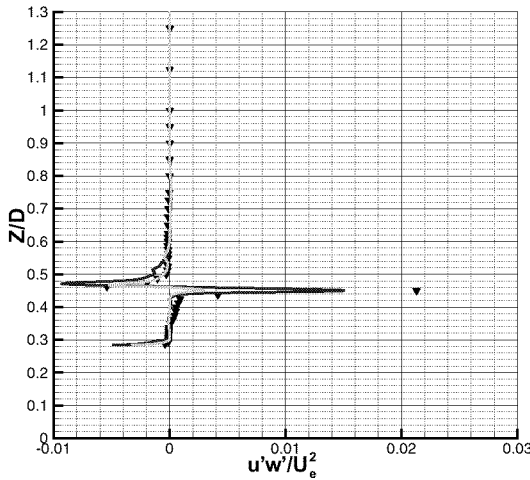The levels of cross-correlation are more satisfactorily predicted by the Launder-Sharma model (see Figs. 18 and 19).

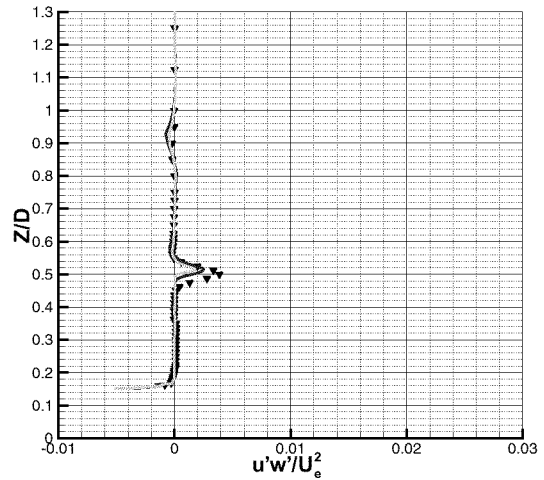Figure 18: Plug nozzle aerodynamics.
Cross correlation profiles at X/D=0.1

Figure 19: Plug nozzle aerodynamics.
Cross correlation profiles at X/D=0.75

4. CONCLUSION

The spectacular increase in our computing capacity during the past 30 years led to a certain despise of the experimental activity. It was anticipated that "numerical wind tunnels" will soon replace the noisy, difficult to operate, dangerous and costly real wind tunnels. This is not our purpose to enter into this polemics. It would not be wise, by reaction, to despise the computational activity which has taken a considerable place in the design and development of nearly all the industrial products (and in many other sectors too!). Because of the technical and scientific difficulties encountered in the domain and the necessity to rely on safe methods, the aerospace industry has strongly invested in the development of codes since the beginning

of the CFD era. Many of the numerical schemes have been devised to improve calculation methods for fluid mechanics applications, notably in the field of aerodynamics.

However, the development of more and more performant codes has not completely killed the experimental activity. Rather soon, it appeared that the confidence in code predictions was limited, the cause of the shortcomings being due partly to uncertainties in the numerical handling of the equations, partly to the lack of accuracy and representativity of the physical laws - or models - implemented in the codes. This perception of the reality has motivated a renewal of interest for experiments since only the confrontation with experimental data can validate - or invalidate - a code. To validate their codes, numericians need an as complete as possible information on some representative test cases. This information constitutes what is called a data bank which must respect certain rules to be useful. Thus, the data bank must contain a precise description of the configuration, along with all the necessary flow and boundary conditions. The measurements must be considered as safe, and if possible accurate. A great accuracy is not always mandatory (it costs much money), but uncertainty margins must be given.

Constitution of a data bank is not a straightforward operation. In addition of technical skill to fabricate a test set up, to operate the wind tunnel and execute the experiment, to perform the measurements, it requires a solid background in fundamental fluid mechanics. The data bank constitution is not limited to the acquisition of a vast amount of results, but must be accompanied by an in depth analysis of the flow physics. Because of the investment needed by such operations and their strategic importance for the development of predictive methods, the question of the data bank dissemination inevitably arises. It is now realized that a good data bank can be as precious as a code and cannot be freely transmitted. Even basic experiments have now an economic weight and cannot be put on the market without something in exchange. Thus, dissemination rules have to be more precisely defined according to the more or less precious nature of the data bank treasure.

In addition of the permanent scientific concern about more accurate, safer and less expansive predictive methods, the problem of the constitution of valuable, safe, well identified and permanent data banks is now considered as a strategic issue and addressed seriously. In this perspective, the Onera Fluid Mechanics and Energetic Branch has started the constitution of a data bank containing the most prominent experimental results obtained in its research wind tunnels of the Chalais-Meudon Center over the last 30 last years (Benay, 2001). This task will be actively pursued and the data bank contents fed with new experiments satisfying the quality criteria here above defined.

5. ACKNOWLEDGEMENT

6. REFERENCES

Benay, R. (2001) La base de données du DAFE. Mise à jour 2001 (The data bank of the Fundamental/Experimental Aerodynamics Department. Update 2001). Onera Rapport Technique N° RT 3/03589 DAFE, juillet 2001

Benay, R. and Servel, P.(1995) Applications d'un code Navier-Stokes au calcul d'écoulements d'arrière-corps de missiles ou d'avions (Application of a Navier-Stokes code to the calculation of flow past missile and aircraft afterbodies), *La Recherche Aérospatiale*, No. 6, pp. 405-426 (1995)

Benay, R. and Servel, P. (2001) Two-equation [k-σ] turbulence model: Application to a supersonic base flow, *AIAA Journal*, Vol. 39, No. 3, 2001, pp. 407-416

Bohning, R. and Doerffer, P. (1997) Passive control of shock wave/boundary layer interaction. In *Notes on Numerical Fluid Mechanics*, Vol. 56, Vieweg, 1997

Bur, R., Corbel, B., and Délery, J. (1998) Study of passive control in a transonic shock wave/boundary layer interaction. *AIAA Journal*, Vol. 36, No. 3, 1998, pp. 394-400

Borrel, M., Montagne, J.-L., Diet, J., Guillen, Ph. and Lordon J. (1988) Calculs d'écoulements supersoniques autour de missiles tactiques à l'aide d'un schéma décentré (Calculation of the supersonic flows past tactical missiles with a centered scheme). *La Recherche Aérospatiale*, pp. 2-20 (1988)

Chanetz, B., Benay, R., Bousquet, J.-M., Bur, R., Pot, T., Grasso, F. and Moss, J. (1998) Experimental and numerical study of the laminar separation in hypersonic flow, *Aerospace Science and Technology*, No. 3, pp. 205-218 (1998)

Chanetz, B., Bur, R., Pot, T., Pigache, D., Gorchakova, N., Moss, J. and Schulte, D.(1999) Shock wave/boundary layer interactions in low density flow: comparisons between flow field measurements and numerical results, *21st Int. Symposium on Rarefied Gas Dynamics - Vol. 2*, Brun, R., Campargue, R., Gatignol, R., Lengrand J.-C. Editors, Cepadues Eds., Toulouse, France, July 1999, pp. 537-544

Chien, K. Y.(1982) Prediction of channel and boundary-layer flows with a low-Reynolds-number turbulence model. *AIAA Journal*, Vol. 20, No. 1, 1982, pp. 33-38.

Délery, J. (1984) Shock-wave / turbulent boundary-layer interaction and its control. Edited by A.D. Young, *Prog. Aerospace Science*, Vol. 22, No. 4, 1985, pp. 209-280 ; also ONERA TP-1984-27, 1984

Délery, J. and Sirieix, M. (1979) Ecoulements de culot (Base flows). AGARD LS-98 on *Missile aerodynamics*, Ankara, Rome, Bruxelles, 5-16 March 1979, Onera T.P. n° 1979-14F

Espina, P.I. and Piomelli, U. (1997) Validation of the NPARC code in supersonic base flows". AIAA Paper 97-0032, Jan. 1997

Fureby, C., Nilsson, Y. and Andersson, K. (1999) Large eddy simulation of supersonic base flow. AIAA Paper 99-0426

Grasso, F. and Marini, M. (1996) Analysis of hypersonic shock wave / boundary layer interaction phenomena. *Computers and Fluids*, 25, pp. 1-21 (1996)

Herrin, J.L. and Dutton, J.C. (1993) Supersonic base flow experiments in the near-wake of a cylindrical afterbody. AIAA Paper 93-2924

Launder, B. E. and Sharma, B. I. (1974) Application of the energy dissipation model of turbulence to the calculation of flows near a spinning disc. *Heat and Mass Transfer*, Vol. 1, 1974, pp. 131-138

Moss, J. and Olejniczack, J. (1998) Shock wave/boundary layer interactions in hypersonic low density flows. AIAA Paper 98-2668

Moss, J.N., Pot, T., Chanetz, B., and Lefebvre, M. (1999) DSMC Simulation of shock/shock interactions: emphasis on Type IV interactions. *22nd International Symposium on Shock-Waves*, London, UK, July 18-23, 1999, Paper 3570

Poll, D. I. A., Danks, M., and Humphreys, B. E. (1992) The aerodynamic performance of laser drilled sheets. *Proceedings of the First European Forum on Laminar Flow Technology*, Hamburg, Germany, 1992, pp. 274-277

Reijasse, P., and Corbel, B. (1997) Basic experiments on non-adaptation phenomena in aerospike nozzles. AIAA Paper 97-2303, June 1997

Sahu, J. (1992) Numerical computations of supersonic base flow with special emphasis on turbulence modeling. AIAA Paper 92-4352

Sauter, J.M. and Dutton, J.C. (1989) Design of an axisymmetric supersonic wind tunnel and experimental study of supersonic, power-off base flow phenomena. Dept. of Mech. and Ind. Eng., Univ. of Illinois at Urbana-Champaign, UILU ENG 89-4002, Urbana, IL, March 1989

Schulte, D., Henckels, A. and Wepler, U.(1998) Reduction of shock-induced boundary layer separation in hypersonic inlets using bleed. *Aerospace Science and Technology*, 2, No. 4 (1998)

Stanewsky, E., Délery, J., Fulker, J., and Geissler, W. (Eds., 1997) *Euroshock - Drag reduction by passive shock control*. Notes on Numerical Fluid Mechanics, Vol. 56, Vieweg, 1997

Tucker, P.K. and Shyy, W.(1993) A numerical analysis of supersonic flow over an axisymmetric afterbody. AIAA Paper 93-2347

Yakhot, V. and Orszag S.A.(1986) Renormalisation Group Analysis of Turbulence. I. Basic Theory, *Journal of Scientific Computing*, Vol. 1, No. 1, 1986, pp. 3-51